

# Simplified Woven Fabric Geometry for Tensile and Bending Analysis

Ungku Nur Ainaa Ungku Mohd Aziz<sup>1</sup>, Suzaini Abdul Ghani<sup>1</sup> and Mohamad Faizul Yahya<sup>1\*</sup>

<sup>1</sup>*Faculty of Applied Sciences, Universiti Teknologi MARA, 40450 Shah Alam, Selangor, Malaysia*

\*Corresponding author's e-mail: [mfy@uitm.edu.my](mailto:mfy@uitm.edu.my)

*Received: 04 June 2021*

*Accepted: 13 January 2022*

*Online First: 02 March 2022*

## ABSTRACT

*This paper introduced a simplified finite element analysis (FEA) tensile model to replace the actual prototype with the FEA software. FEA was performed using commercially available ABAQUS software. FEA is a computerized method of predicting how a product reacts to real-world forces, vibrations, heat, fluid flow, and other physical effects. Furthermore, fabric performance can expect by using FEA in textile testing. This project aims to determine the tensile and bending analysis of the plain-woven fabric using ABAQUS software. The fibre modulus was configured 10 % and 20 % lower than the initially configured modulus. The mesh size is reduced to 20 % and 40 % to determine the stress and strain curve and analyze the bending model. The results showed that the stress-strain curve would be affected as the modulus and mesh size decreases.*

*Keywords: Tensile; bending; finite element analysis; stress; strain*



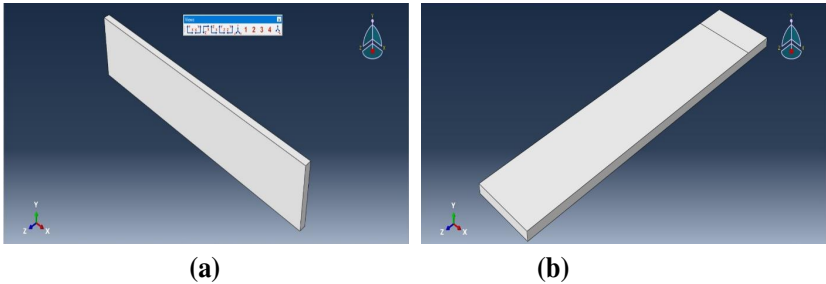
## INTRODUCTION

The finite element analysis (FEA) is a simulation of any given physical phenomenon using a numeric technique known as the finite element method (FEM). Engineers use FEA software to reduce the number of physical prototypes and experiments and optimize components in their design phase to develop better products faster while saving on costs. Finite element analysis (FEA) is also widely used by engineers in different industries. According to Mills [1], FEA is commonly used in the automotive sector to design car bodies and model occupant protection. It is also used to replace the actual prototype, thus lowering the real prototype's cost. Finite element analysis in textile allows mechanical properties to be simulated before actual production. The FEA enable the modelling and representation of materials by considering various geometric parameters such as yarn undulation, the probability of symmetries in the unit cell and the interaction between warp and fill tows. These factors help generate a mesh close to the actual fabric or composite [2].

A finite element analysis requirement is constructing an accurate geometric model of the object. Many methods and concepts for developing a 2D or 3D model using CAD have been proposed. The structural simulation for the fabric's geometry is solidified to provide the basis for the composites' mechanical analysis based on the material [3]. It can also generate different geometric patterns of fabrics such as woven, woven, and knitted fabrics. The linear static analysis applied to most FEA models follows Hooke's elasticity law. Hooke's law is the first classical example of an explanation of elasticity, which is the property of an object or material that allows it to be restored to its original shape after being distorted. Young's modulus or elastic modulus is essentially the slope of the straight flexible part of the stress-strain curve. In other words, it is known as a ratio of tensile stress to tensile strain. It defines the relationship between stress (force per unit area) and strain (proportional deformation). These properties represent material strength and how a structure deforms when subjected to various loads. The study aims to determine the tensile and bending analysis of the woven fabric using ABAQUS software. ABAQUS is known as one software used to produce FEA models. It is widely available in universities by providing detailed explanations of foam models. Students can also use up to 1000 designs.

## METHODOLOGY

This methodology explained the flow of the process that is used in this tensile model. Furthermore, there are a few illustrations and table that are provided below.



**Figure 1: (a) The model before the partition is inserted (b) The model after the partition is inserted**

Figure 1(a) illustrates the tensile model outcome after the plate size value has been inserted with 20 mm in length, 4 mm in width, and 0.5 mm in thickness. The tensile plate has, therefore, been created with the value given. However, the partition also has been made with 2 mm length, as shown in Figure 1(b). The partition is needed to provide the bending effects to the material.

### Material Property

The following table shows the value of the material properties required in this tensile model for analysis. Various material databases are obtained from material testing.

**Table 1: The value of density, elastic, and plastic that have been inserted in the tensile model**

Density	Material Property			
	Elastic		Plastic	
	Young Modulus (MPa)	Poisson Ratio	Yield Stress (N/m <sup>2</sup> )	Poisson Strain
1.41x10 <sup>-9</sup>	10861	0.3	0.1	0.0
			200	0.1
			350	0.2
			420	0.3

In the second step, three properties are inserted: density, elastic, and plastic with a specific value, as shown in Table 1. The section manager named Section-1 has been added. Next, the step was repeated and changed the elastic properties in Table 2, as shown. Young's modulus has been reduced to 10% and 20%, respectively.

The third step is to assemble part for a single visualization. This process must proceed to the next step. The instance has been created from parts of it. Part of the tensioning plate has been applied. The type of instance used depends on the part of the mesh. Next, the field output request manager and the history output request manager have also been created.

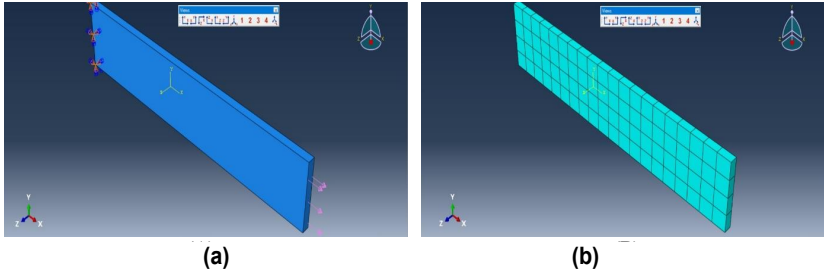
**Table 2: The value of different modulus that has been inserted in the tensile model**

Elastic		
Value	Value Young Modulus (MPa)	Poisson Ratio
Initial	10681	0.3
Less than 10 %	9774.9	0.3
Less than 20 %	8688.8	0.3

**Boundary Condition, Load and Mesh**

For the fifth step in Figure 2(a), boundary conditions have been created, and the type of constraint that has been made is ENCASTRE (J1 = U2 = U3 = UR1 = UR2 = UR3 = 0), which means that it is at all directions and rotational angles. The function is to lock the surface away from moving. Next, the load is created to increase the other side's pressure. Lastly, the magnitude

is given at 200 to make it move upwards in the X direction in Figure 2(a). Meanwhile, for the partition, the load is provided on the partition made, and the magnitude is 0.04 to make it move down to evaluate the bending to the model. The following illustration and table show the image and value that have been inserted for the tensile model.



**Figure 2: (a) The model when the load is inserted (b) The model when the mesh is inserted**

**Table 3: Value of different mesh size**

Mesh Size		
Value	Number of Element	Seed size
Initial	24	1.0
Less than 20 %	45	0.8
Less than 40 %	80	0.6

For the sixth step, Figure 2(b), to make a mesh, the mesh size is 1. The hexagon shape is chosen for the mesh shape. According to [4], calculating the accuracy of a hexahedral element is higher than that of a tetrahedral element. Hence, the curve and surface boundary of a higher-order element can accurately approach the structure boundary. Finally, Figure 3 occurred when the part-instance mesh was made. The step then repeated with different sizes of 0.8 and 0.6, which were reduced to 20 % and 40 %, respectively, as shown in Table 3. The purpose of reducing the mesh size is to get a more accurate result of the tensile material, as the smaller the mesh, the less the error. According to Silva & Cardoso [5], finer meshes can provide a more accurate solution. However, as a mesh fineness increases, computational time increases, thus a good balance between accuracy and available computational resources must be executed. After the mesh have been inserted into the model, the colour of the model will automatically change to green which showed that the mesh had been inserted.

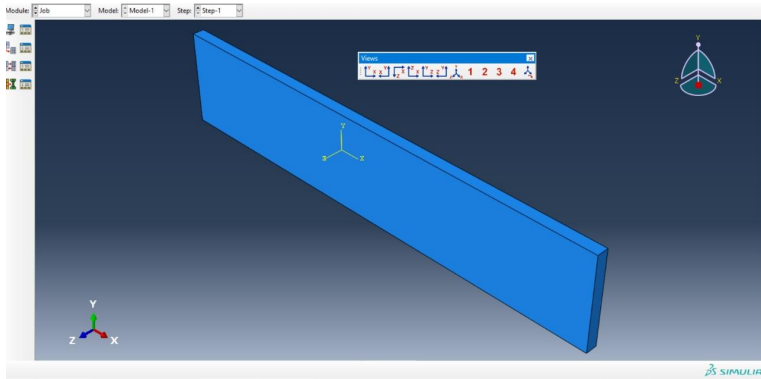


Figure 3: The final visualization of the model

For the last step, the Job Manager was created as a Job-1 and submitted for analysis. Job-1 is finally completed successfully, and the result can be visualized, as shown in Figure 4 below. If there is an error in job submission, the user can edit the data until they can successfully submit it.

## RESULTS AND DISCUSSIONS

### Visualization

Figure 4 showed the result of the successful job submitted. The model is also run successfully. The model is made based on the assumption of the fabric appropriate properties, which is plain-woven fabric. Figure 4(a) showed using the standard coordinate system. Meanwhile, Figure 4(b) showed that the objective is achieved by changing the coordinate system.

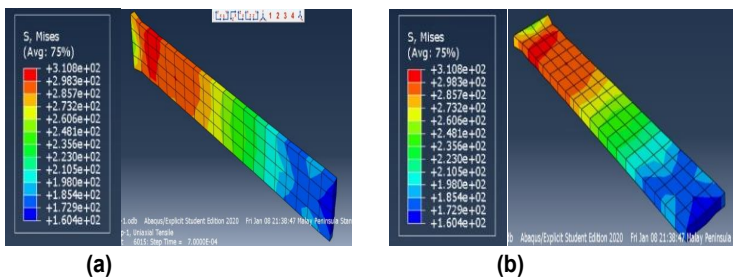


Figure 4: Visualization of the FEA model set-up with the different coordinate system

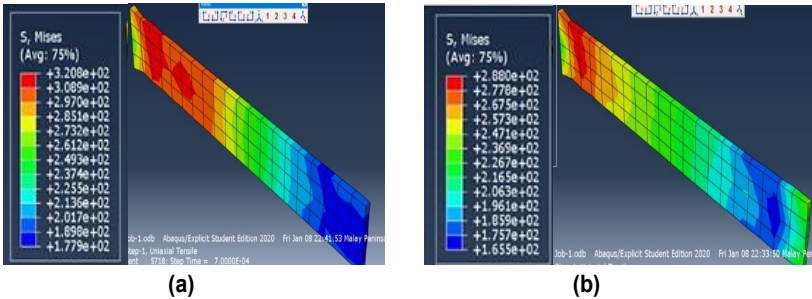


Figure 5: Visualization of modulus that less than 10% (a) and 20% (b)

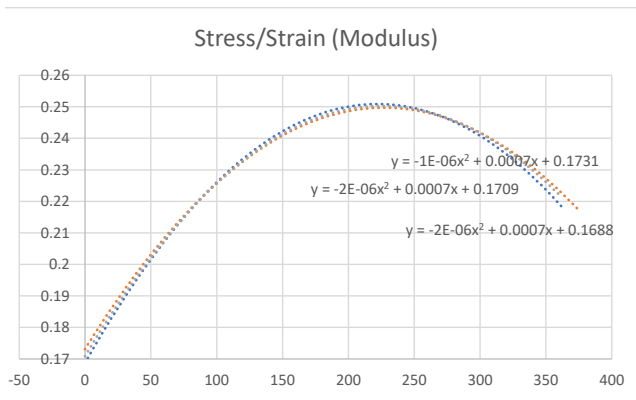


Figure 6: The graph of stress and strain using different modulus

Young modulus is the straight elastic part of the strain curve slope. The blue line represents the initial value of modulus, and the orange line represents the value that less than 10 % from the initial and the grey line represent the value that less than 20 % from the initial. It also described the behaviour of the material based on the graph shown. The material's stiffness is also essential and can easily bend or stretch. The reducing modulus value is based on Table 2, as mentioned above. Based on Figure 5(a), there is more stress at the boundary area as the surface is locked away, but the pressure is reduced as the modulus is reduced to 10 %. Figure 5(b) also showed that stress is reduced. Hence, when a material reached specific pressure, the material will begin to deform. As a result, it can be concluded that as the modulus was reducing, the stress also was reduced. It is shown that reduction of modulus would lead to lower stress results.

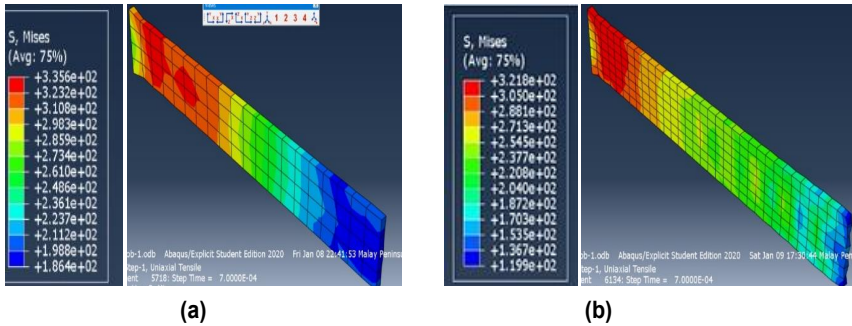


Figure 7: Visualization of reducing mesh size that less than (a) 20% and (b) 40%

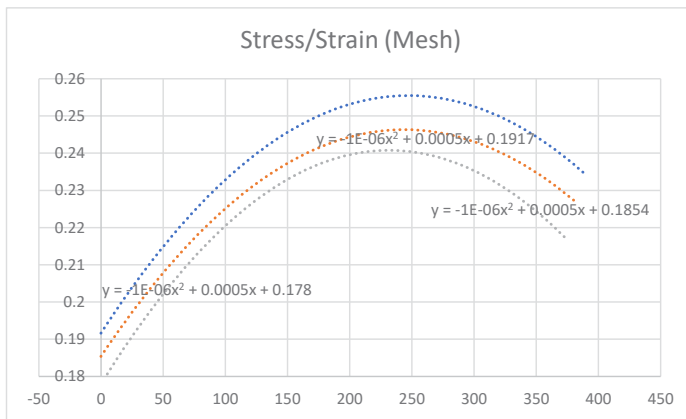
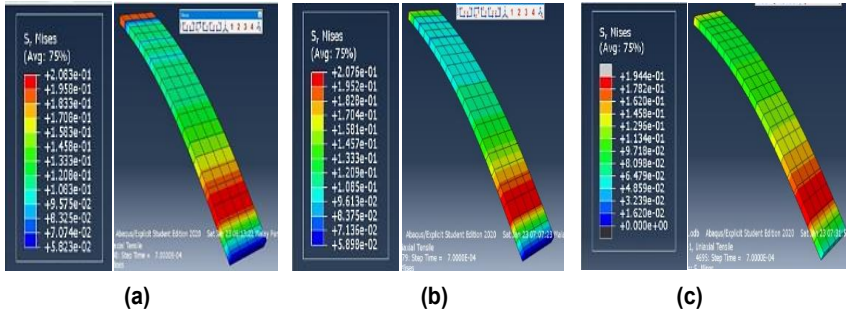


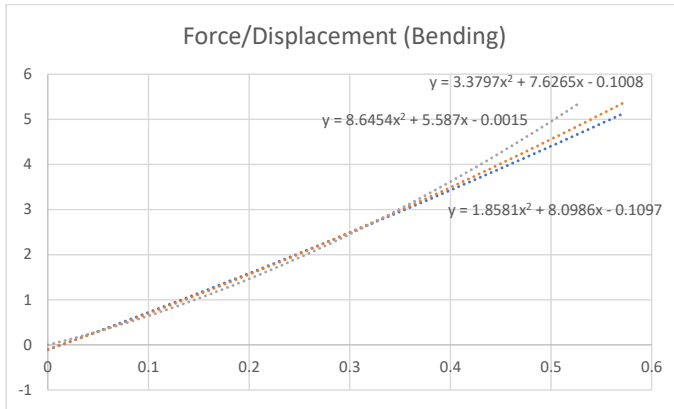
Figure 8: The graph of stress and strain using different mesh size

In FEA, a mesh is created to divide the domain into a discrete number of elements for calculating the solution. The accuracy of the results can be determined by mesh. The detail of the mesh properties used in the work is outlined in Table 3. This represents the elastic area of the model. Based on Figures 7(a) and 7(b), it is shown that the mesh size decreased to 20% and 40%, the total element increased, and the stress on the boundary area increased from the initial in Figure 4(a). As the mesh density increases, the quantity of interest begins to converge to a particular value and vice versa. However, the graph in Figure 8 shows that stress and strain decrease as the mesh size decreases. The blue line represents the initial value, the orange line represents 20% less than the initial value, and the grey line represents 40% less than the initial value.





**Figure 9: Visualization of bending with a different modulus that less than 10% and 20% from initial**



**Figure 10: The graph of force and displacement in bending using different modulus**

Figure 9 shows the bending of the woven fabric with different modulus. The blue line represents the initial modulus. The orange line represents the modulus value of less than 10% from the initial, and the grey line represents 20% less than the initial modulus. Figure 9(a) used an initial modulus of 10 861 MPa. In the meantime, Figures 9(b) and (c) use different modulus, which is less than 10% and 20% lower than the initial modulus. The visualization showed that the higher the modulus, the higher the material's force to break. As shown in Figure 10, the bending graph is based on displacement and force. The bending is going down due to the load on the partition given.

## **CONCLUSIONS**

In conclusion, to ensure high-quality parts and minimize the cost of designing such components, it is necessary to develop methods for predicting textiles deformation using the FEA. Based on this study, it can conclude that the different modulus and the mesh size will give different results towards the woven fabric as the higher the modulus, the higher the pressure needed for a material to break. Hence, the higher tension required is because, the higher modulus increases the textile materials strength. Therefore, this study objective is achieved by determining the tensile and bending analysis of the woven fabric using ABAQUS software.

## **ACKNOWLEDGEMENT**

The finite element analysis (FEA) software used in work is Abaqus 2000's student version and available for students upon request. The software allows students to create a model and solve the engineering problem up to 1000 nodes. It is available for students to download at <https://edu.3ds.com/en/software/abaqus-student-edition>.

## REFERENCES

- [1] N. Mills, 2007. Chapter 6 - Finite element modelling of foam deformation. *Polymer Foams Handbook*, 115-145.
- [2] H. Ali, A. Hassan, F. Mazhar, (2015). Finite element analysis of mechanical properties of woven composites through a micromechanics model. *Science and Engineering of Composite Materials*.
- [3] Y. Shen, A. J. Meir, Y. Cao, S. Adanur, 2015. Finite Element Analysis of Monofilament Woven Fabrics under Uniaxial Tension. *The Journal of The Textile Institute*, 106(1), 90-100.
- [4] W. Liu, M. Geni, L. Yu, 2011. Effect of Mesh Size of Finite Element Method in Modal Analysis for Periodic Symmetric Struts Support. *Key Engineering Materials*, 462-463, 1008-1012.
- [5] V. B. R. E. Silva, J. Cardoso, 2020. Chapter 2 - How to approach a real CFD problem—A decision-making process for gasification. *Computational Fluid Dynamics Applied to Waste-to-Energy Processes*, 29-83.